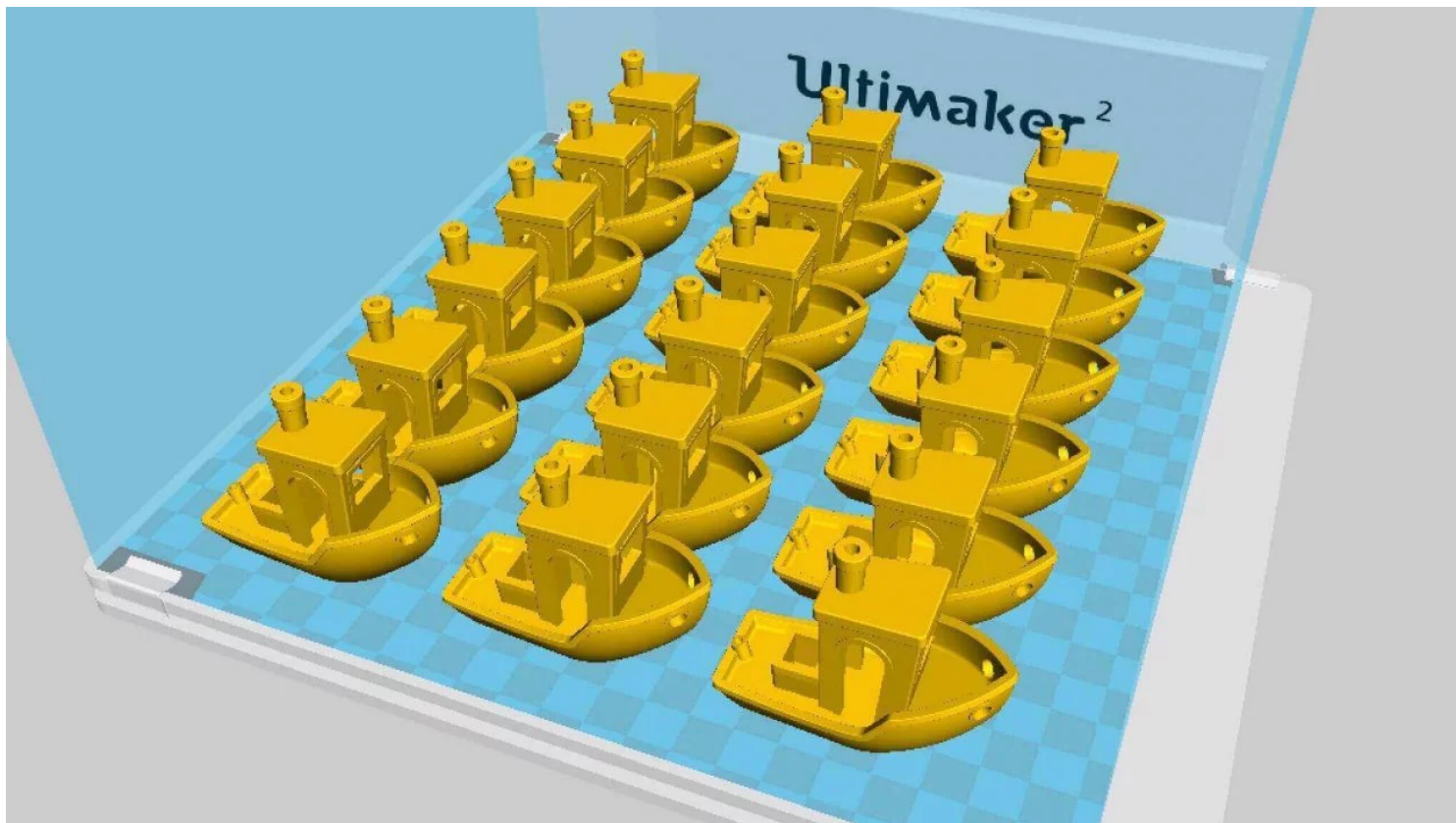


EXHIBIT

35



Tips and Tricks

3D Slicer Settings for Beginners – 8 Things You Need to Know



by All3DP

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

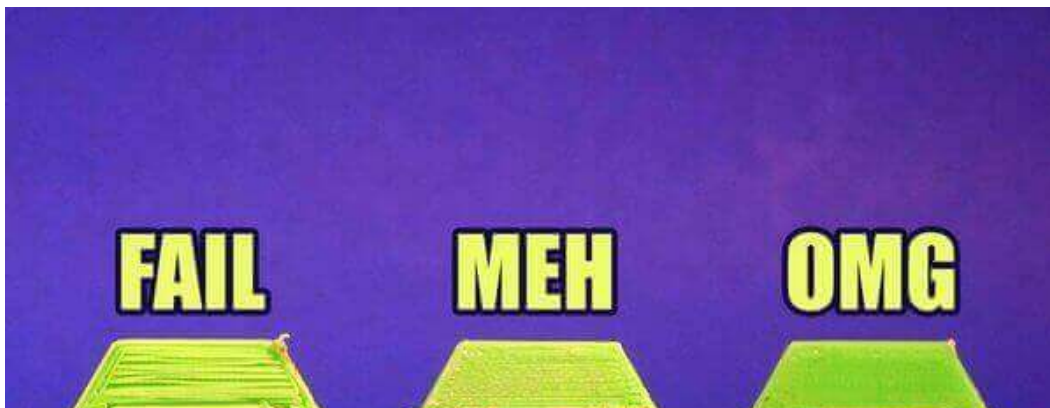
[BACK TO TOP](#) ^

Learn about best practices for 3D slicer settings in this guest post from 3D printing community Pinshape. Slice like a champ and get better prints!

Guest Post: This article originally appeared on the [Pinshape Blog](#). Text and images reproduced with kind permission of the team at [Pinshape](#).

Proper 3D slicer settings can mean the difference between a successful print, and a failed print. That's why it's so important to know how slicers work and how each different setting will affect your results.

We understand that the many settings on slicing software can be intimidating, especially for beginner makers. Sometimes even advanced makers make mistakes and end up with failed prints. Just ask Pinshaper & experienced 3D printer, [Zheng3!](#) His picture below illustrates a simple but effective example of the difference that 3D slicer settings can have on a print.



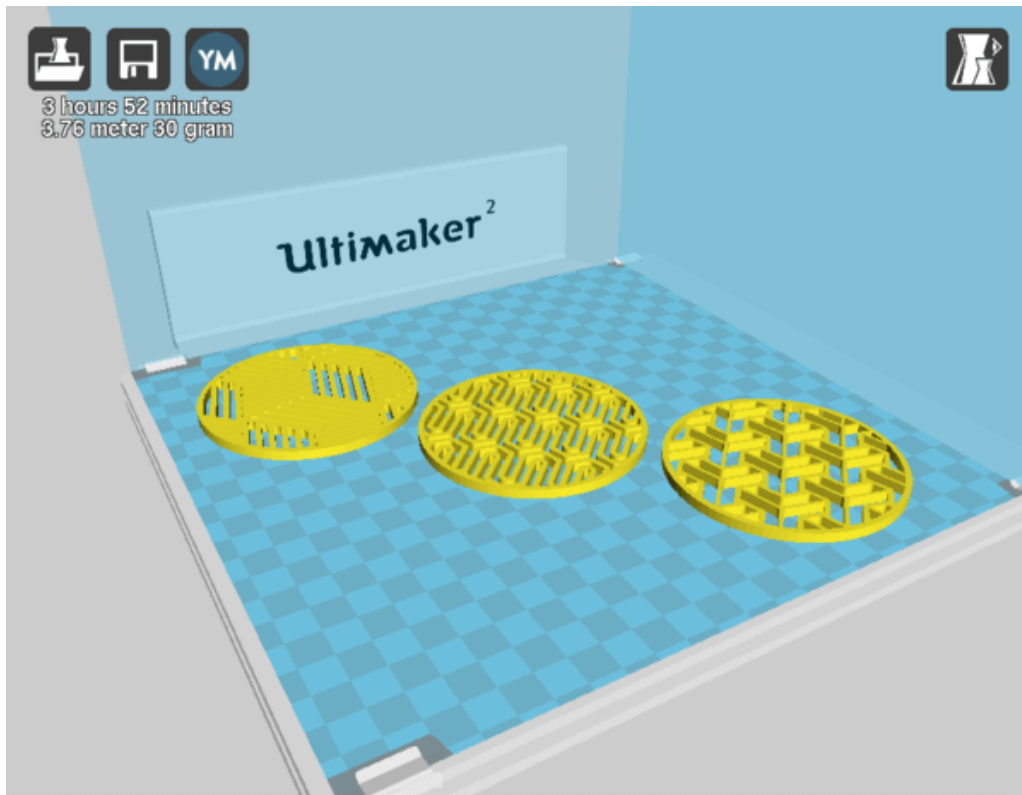
This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

[GOT IT.](#)



Part of the problem is that the optimal slicer settings depend on what design you're printing and what material you're using, so there is no "one setting fits all" perfect setting. The big question, then, is: **how do you know what slicer settings to use on which designs & material?**

To break it down, let's go through some of the basic features of a slicer, and talk about how each setting will affect your print. This is more of an introduction to the topic than an in-depth guide.



What Is a 3D Slicer & What Does It Do?

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

code file which is sent to your printer. Slicer settings do impact the quality of your print so it's important to have the right software and settings to get you the best quality print possible.

For the examples, we will use Cura (version 15.04.3), a free slicer with similar features to most other slicers.

The basic settings menu in an older version of Cura looks like this:

8 Slicer Settings You Need to Know & How They Work!

1. Layer Height

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

that make up your object.

If you're printing something without detail, a thicker layer will get you a faster print but it will be a rougher surface and the individual layers will be more visible. Low resolution printing is good for things like prototyping where details may not be necessary.

If you want to print something with intricate details, you will get the best print with a thinner layer height. Cura recommends settings of .06mm for a high resolution print like this [Tudor Rose Box by Louise Driggers](#)

EDIT: After consulting with a few of our community makers, we found that a layer height of .06mm is not a realistic setting for most FDM printers. Here is what one of our pro makers [Dan Steele](#) recommends for detailed settings:

.4mm nozzle fine = .1mm average=.2mm rough=.34mm

.35mm nozzle fine= ,1mm avg = .2mm rough = .3mm

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

of detail like this [Spiral Chess Set by BigBadBison](#). This is the layer height we use as our go to in the Pinshape office on our Ultimaker 2.

Larger layers work best for prints that don't have a lot of detail. Cura recommends .2mm for a "low resolution" print with little detail like this [Elephant by le FabShop](#).

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

PRO TIP: 3D printing veteran [Chris Halliday](#) recommends changing one setting at a time, keeping track of how each incremental change affects your print!

2. Shell Thickness

Shells refers to the number of times the outer walls of the design are traced by the 3D printer before starting the hollow inner sections of your design. This defines the thickness of the side walls and is one of the biggest factors in the strength of your print. Increasing this number will create thicker walls and improve the strength of the print. It is automatically set to .8 so there shouldn't be any reason to change this for decorative prints. If you print something that will need more durability, or if you're creating a water-tight print like a vase, you may want to increase shell thickness.

3. Retraction

This feature tells the printer to pull the filament back from the nozzle and stop extruding filament when there are discontinuous surfaces in your print, like this one:

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

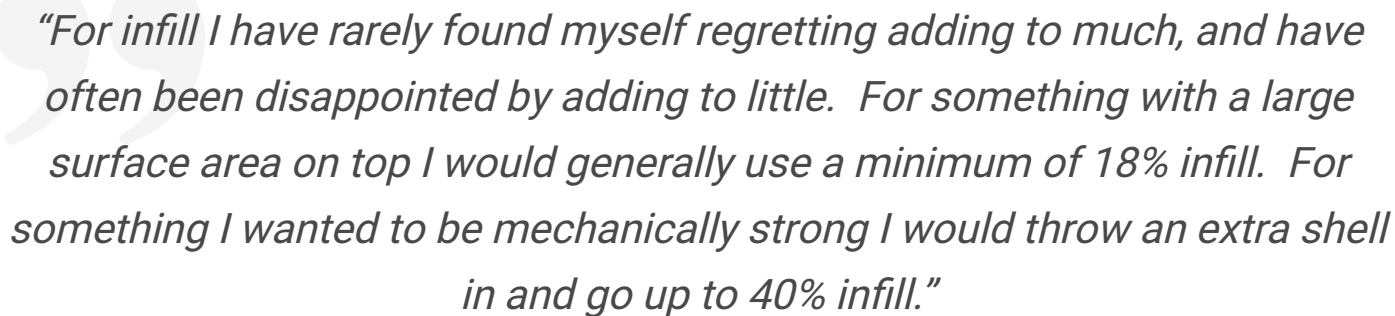
Retraction is usually always enabled, unless your print doesn't have any discontinuous surfaces in it. This setting can sometimes cause filament to get clogged in your nozzle during a print in which case you probably want to disable it. If you find there is too much filament oozing out of the nozzle, leaving your print with a bunch of strings or clumps on the outer edges, then be sure to turn on retraction.

4. Fill Density

Infill refers to the density of the space inside the outer shell of an object. You'll notice this is measured in % instead of mm like the layer height. If an object is printed with 100% infill, it will be completely solid on the inside. The higher the percentage of infill, the stronger and heavier the object will be, and the more time and filament it will take to print. This can get expensive and time consuming if you're printing with 100% infill every time – so keep in mind what you'll be using your print for.

If you're creating an item for display, 10-20% infill is recommended. If you need something that is going to be more functional and sturdy, 75-100% infill is more appropriate. Cura infill creates a grid like pattern inside your object which gives the top layers of your model more support.

One of our community members, [Dan Steele](#) is a fan of more infill than less:



“For infill I have rarely found myself regretting adding to much, and have often been disappointed by adding to little. For something with a large surface area on top I would generally use a minimum of 18% infill. For something I wanted to be mechanically strong I would throw an extra shell in and go up to 40% infill.”

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

5. Print Speed

Print speed refers to the speed at which the extruder travels while it lays down filament. Optimal settings depend on what design you're printing, the filament you're using, the printer, and your layer height. Of course, everyone wants to print their object as quickly as possible, but fast print speeds can cause complications and messy looking prints.

For complicated prints, a slower speed will give you a higher quality print. A good starting point that Cura recommends is 50mm/s. You can also play around with speed and see what works best for your printer.

6. Supports

Supports are structures that help hold up 3D objects that don't have enough base material to build off of as they are being printed. Since objects are printed in layers, parts of an object that extend past a 45 degree angle will have nothing for the first layer of filament to build on.

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

1. Anything in a "T" shape is safe to print without support because it's a gradual slope which still has enough material beneath it to keep it from drooping. This is another way to think of the **45 Degree Rule**, which states that in general, overhangs with a slope greater than 45 degrees will require supports.
2. Designs that take the form of an "H", where the middle overhang connects to either side is called bridging. Any type of bridge should have supports to prevent drooping or a messy print.
3. Anything with a "T" shaped overhang will need support to avoid drooping.

SUPPORT TYPE

In the drop down menu, there are two types of support you can choose from:

- Touching Build Plate – this is for designs where the section of the design that needs the support can attach to the build plate like this:

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

go in between parts of the design and touch the top of the model.

7. Platform Adhesion Type

These settings will affect how your model sticks to the print bed. Warping at the bottom of a design can be a main culprit for prints not sticking to a print bed, but there are two main settings you can adjust to help with platform adhesion:

- Raft: A horizontal grid that goes under the object that acts as a platform to stick to the bed and build from. They can also be useful when printing models with small parts at the bottom of your print, like animal feet. If you do choose to use a raft, it will leave rough edges on the bottom of your print when you remove it.

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

- **Brim:** Like a brim of a hat, brims are lines around the bottom of the object which keep the corners of your model down without leaving marks on the bottom of the object. This is a better option if your main objective is to get your model to stick to the print bed. Brims can also be used to stabilize delicate parts of an object that are isolated from the rest of the model like the legs of a table.

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

8. Initial layer thickness

This is located in advanced settings in Cura and refers to the thickness of your very first layer on the print bed. If you want a more sturdy base for your print, you can make the initial layer thicker. The default on Cura is .3mm which gives a thick bottom layer that's easy to build on and sticks to the platform well.

What's the difference between initial layer thickness and bottom/top thickness in the basic settings? While the initial layer thickness is the very first layer that goes down, the bottom and top thickness refers to how many mm of solid material will be set down before your infill is created.

These are the basic settings for a slicer program – if you want to get into more advanced territory, there are more settings but these are the main ones a beginner needs to be aware of.

PRO TIP: When venturing into more complicated prints, 3D printing pro [Zheng3](#) has a few steps to add on to Chris Halliday's advice on changing one setting at a time:

1. **Write down all your settings.** Label these settings as a group with a capital letter. e.g. rex_A, rex_B, rex_C. Screenshots of print settings will be handy here.

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

4 Different 3D Slicer Programs

If you haven't figured out which slicer program works best with your printer, here's some options on the market to get you started:

Cura (Free)

Cura is made by Ultimaker and is extremely user friendly & fast so it's great for beginners. It is not a proprietary software so it works for multiple different printers. The tradeoff of the ease of use is that you have less control over some of the more detailed settings. There are, however lots of plugin options for you to add if you need any of those extra features. [Download Cura](#)

Slic3r (Free)

This is an open source slicing project started by the [RepRap](#) Community & works on multiple printers. Their focus and design goal is ease of use and maintaining the original design. One unique feature is that it allows you to vary the infill pattern across layers which can increase the strength of your print. The user interface has improved dramatically since they just started and it has positive reviews from most of the community. [Download Slic3r](#)

Simplify3D (\$149 USD)

This is one of the paid slicers on the market — so why should you choose to pay when you have so many other options for free? The main point we've heard from the community is

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.


Although there is no free trial version of this software, they do let you return the software within two weeks if you don't like it. If you are a more advanced maker and care about control and speed, the investment might be worth it! [Buy Simplify3D](#)

Makerbot Desktop (Free)

Formerly known as Makerware, the Makerbot slicer software has been rebranded as Makerbot Desktop. The settings are similar to Cura and are very basic and easy to navigate. You can also create custom profiles in this software but there is no user interface for this function so you must use a text editor. Feedback from the community is that it can be very slow compared to alternatives. [You can download this software from the Makerbot website.](#)

PRO TIP: Still need some advice on how to figure out slicers? Here's a great overall tip from a 3D printing expert Richard Horne, compliments of [3D printing for beginners](#):

"Print out lots of 20mm cubes. It's quite a boring object, but it can help ensure you have a well setup and calibrated machine."

 License: The text of "[3D Slicer Settings for Beginners – 8 Things You Need to Know](#)" by [All3DP](#) is licensed under a [Creative Commons Attribution 4.0 International License](#).

SUBSCRIBE

Subscribe to updates from All3DP

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

RECOMMENDED FOR YOU

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

Recommend

Tweet

Share

11/25/2018



Join the discussion...

LOG IN WITH

OR SIGN UP WITH DISQUS ?

Name

The Top Guitarist • 2 months ago

Slic3r is extremely slow compared to cure.

^ | v • Reply • Share ›

Subscribe Add Disqus to your siteAdd DisqusAdd Disqus' Privacy PolicyPrivacy PolicyPrivacy PolicyPrivacy

TOPICS

[CURA](#) [SIMPLIFY3D](#) [SLICING](#) [SOFTWARE](#)

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.

All3DP

World's Leading 3D Printing Magazine with Compelling Content. For Beginners and Pros. Useful, Educational, and Entertaining.

Information

- [ABOUT US](#)
- [PRIVACY POLICY](#)
- [TERMS OF USE](#)
- [IMPRINT](#)
- [JOB OFFERS](#)

Links

- [GET IT 3D PRINTED](#)
- [NEWSLETTER](#)
- [CONTENT ACADEMY](#)
- [ADVERTISE WITH US](#)

Follow us

[FACEBOOK](#)
[TWITTER](#)
[GOOGLE+](#)
[REDDIT](#)

This website or its third-party tools use cookies, which are necessary to its functioning and required to achieve the purposes illustrated in the [Privacy Policy](#).

GOT IT.